Supersonic Retropropulsion

Guy Schauerhamer, *Johnson Space Center*
Kerry Trumble, *Ames Research Center*
William Kleb, *Langley Research Center*
Karl Edquist, *Langley Research Center*
Scott Berry, *Langley Research Center*
Matt Rhode, *Langley Research Center*

As NASA turns its focus to space exploration beyond low-Earth orbit, one clear goal is reaching Earth's neighboring planet—Mars. To meet this goal, a spacecraft must decelerate from tens of thousands of miles per hour to a delicate, soft landing on the Mars surface, which is very challenging. To add complexity to this feat, the mass of future spacecraft is rising.

Since the current robotic missions (approximately 1 metric ton) have already reached the limit of scalability for current deceleration techniques, larger robotic- or human-scaled (10s of metric tons) missions to Mars require a new enabling technology.

Supersonic Retropropulsion (SRP) is a viable means for decelerating high-mass vehicles during Martian atmospheric entry. Retropropulsion has been used successfully in previous missions during the final stages of Martian landing, but its flow characteristics at higher (supersonic) velocities, experienced earlier in the entry trajectory, still require much research.

Ground and flight tests would provide designers with data to predict the flow field around a vehicle. But, because setting up entry conditions in wind tunnels and conducting flight tests on Mars can be difficult and cost-prohibitive, analysts use Computational Fluid Dynamics (CFD) to obtain these data and to develop the SRP method.

The SRP flow structure involves an opposing jet firing into a supersonic flow causing shocks, free shear layers, recirculation, and stagnation regions. The complexity of this interaction stretches current CFD capabilities, and mandates validation of CFD for this type of flow.

Four flow solvers across the agency have been employed to validate CFD for SRP. These codes, OVERFLOW (detailed in this report), FUN3D, DPLR, and US3D have solved SRP problems by simulating cases from historic and recent wind tunnel tests. Through code-to-code and code-to-test comparisons of surface pressure, forces and moments, and shock structure, analysts can validate the CFD and confidently use it for the complex Martian entry problem.

CFD was used to simulate historic wind tunnel experiments like Daso et al (American Institute of Aeronautics and Astronautics [AIAA] 2007-1423).

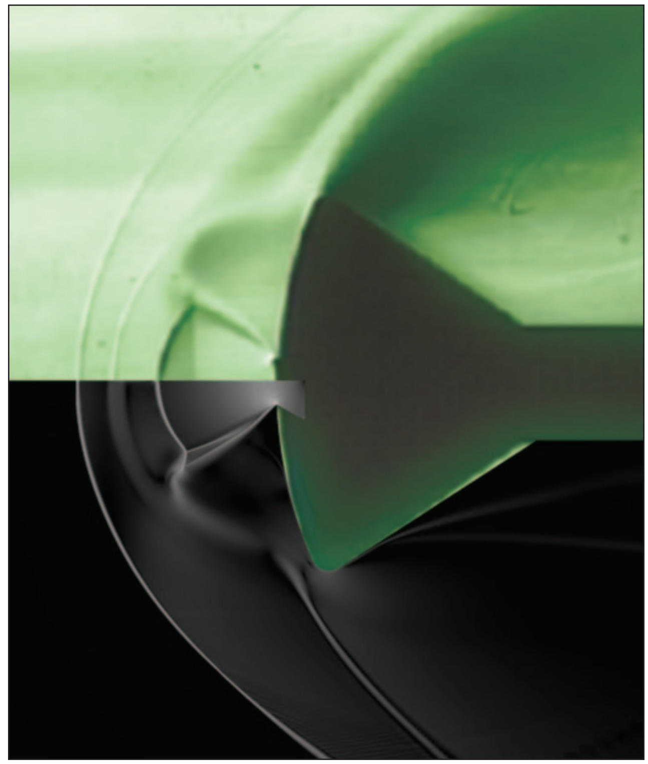

Fig. 1. *Comparison of test Schlieren from Daso et al (top) and the log of the density gradient magnitude from OVERFLOW (bottom).*

Code-to-code and code-to-test comparisons matched closely and were encouraging (figure 1).

Using historic data for SRP development does have its drawbacks—existing documentation lacks critical information needed for CFD validation, and all tested thrust levels were lower than what is desired for flight. Engineers designed a new wind tunnel test specifically for CFD validation to address these concerns.

The team used observations from pretest CFD simulations to improve the design of the test model and run conditions. In CFD results, the original model diameter caused surface pressure discrepancies due to wall effects, low temperatures in the plumes caused concern of liquefaction, possible blockage was shown at high thrust coefficients, and the plume structure contained high-frequency unsteadiness.

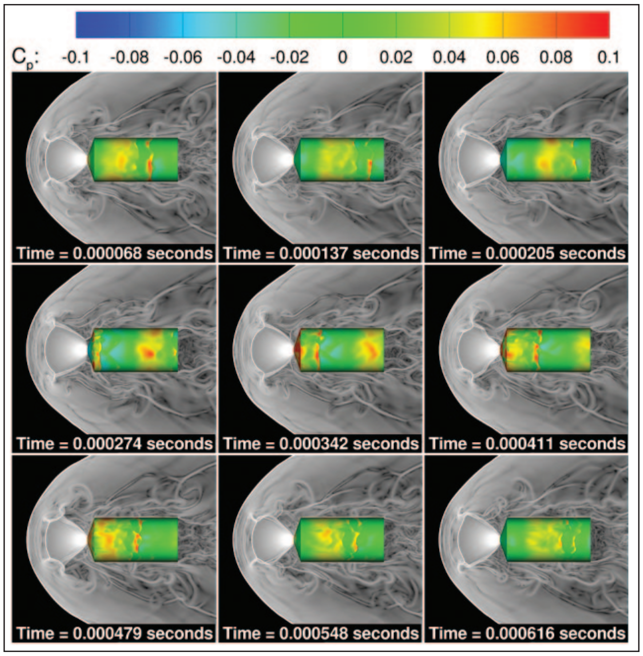

Fig. 2. *OVERFLOW time-accurate simulation of a single nozzle Langley Unitary Plan Wind Tunnel posttest case. The log of the density gradient magnitude are in grayscale and surface pressure coefficient are in color. Mach = 4.6, Reynolds/foot = 1.5E+06, and thrust coefficient = 2.*

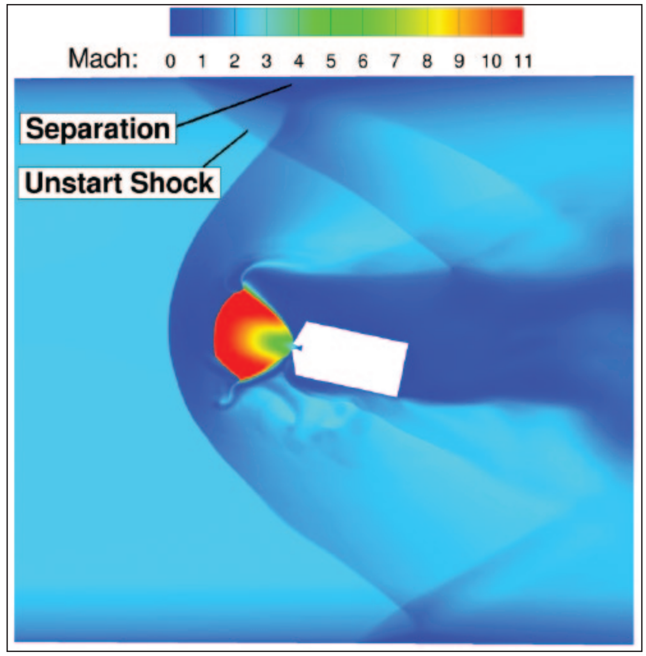

Fig. 3. *OVERFLOW simulation of tunnel unstart in the Langley Unitary Plan Wind Tunnel. Mach = 2.4, Reynolds/foot = 1E+06, thrust coefficient = 4, angle of attack = 12 degrees.*

In response to these observations, the team decreased the test model diameter, heated the plume gas before entering the model plenum, decreased the thrust levels in the run envelope, and added high-frequency pressure gauges and high-speed Schlieren (photographing the flow of air around objects) capabilities.

The team successfully completed the test in July 2010 in the 1.2 x 1.2 m (4 x 4-ft) supersonic Langley Unitary Plan Wind Tunnel. High-speed Schlieren, 165 pressure taps, and 11 high-frequency gauges were employed on a large run matrix to provide the team with the data needed to perform CFD validation.

Each code completed time-accurate sensitivity studies to compare grid refinement, numerical method choice, turbulence model, and time-step values. Once the team members established best practices, they compared final products to tunnel data. This not only supplied CFD validation, but also tested each code's predictive capability.

The test captured high-frequency—approximately 2 kilohertz—unsteadiness with enough detail to use it as a metric for CFD validation. Figure 2 shows an example of a run simulated with OVERFLOW organized in a time series to visualize the unsteady behavior of the jet plume. The flow structure, behavior, dominant frequency, and averaged surface pressures matched those of the test. Similar code-to-test comparison successes for OVERFLOW occurred across the run matrix, with variance in nozzle quantity (1, 3, or 4), angle of attack, and thrust coefficient.

Qualitatively, the code-to-code comparisons differed mostly in the level of unsteadiness, which was dampened by some of the flow solvers. These differences were most likely caused by turbulence model implementation.

At certain conditions, the Langley Unitary Plan Wind Tunnel test showed tunnel unstart—a supersonic choking reaction caused by the bow shock reflection off the wall. At the reflection, a separation region forms and increases in size, causing a new shock that propagates upstream. Unstart was predicted (figure 3) by using OVERFLOW, and modeling the settling chamber, nozzle, and full test section of the tunnel to properly form the large boundary layer on the walls around the model. This capability will help in the design of future SRP wind tunnel tests.

The Langley Unitary Plan Wind Tunnel test provided valuable information for validation of CFD for SRP. However, due to the large boundary layers in the tunnel, test runs could not simulate the high level of thrust needed for flight conditions. The lower thrust coefficients in the test created a fair amount of unsteadiness, which caused concern for vehicle stability. But when higher thrust coefficients were tested, the flow field became much

Supersonic Retropropulsion

continued

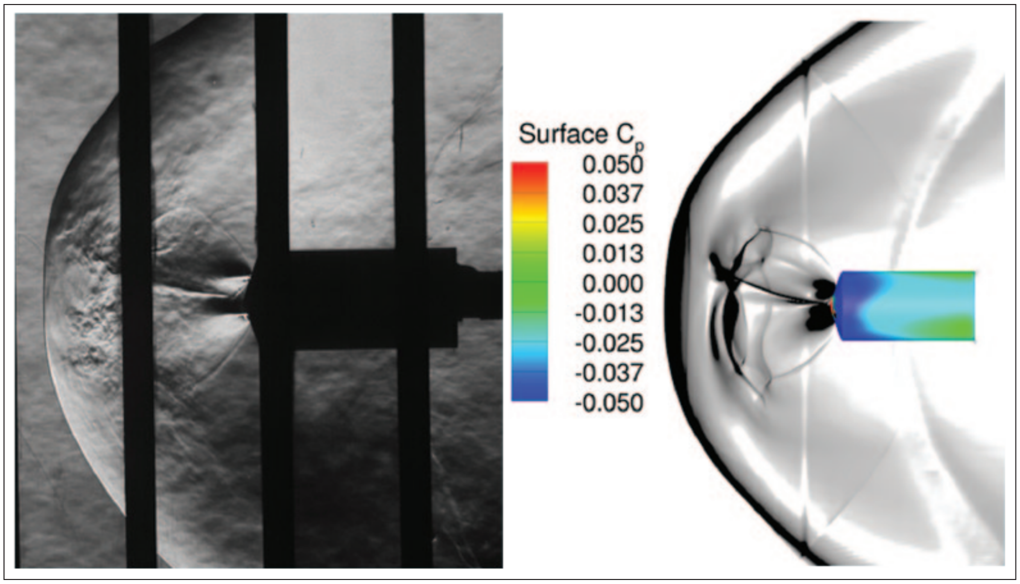

Fig. 4. *Langley Unitary Plan Wind Tunnel triple-nozzle case. Left image is the test Schlieren and the right image is an OVERFLOW simulation. Grayscale coloring displays a constructed Schlieren (in three-dimensional view) of the Computational Fluid Dynamics solution created by a program written by David Saunders; the vertical line through the plume is a grid effect. Mach=3.5, Reynolds/foot=1E+06, thrust coefficient=6.*

steadier. For example, a triple nozzle case at zero angle of attack was much steadier at a thrust coefficient of 6 (figure 4) than at 3.

SRP analysts will use the same model from the Langley Unitary Plan Wind Tunnel test for future analysis in the 2.7 x 2.1-m (9-ft x 7-ft) Ames Research Center Unitary Plan Wind Tunnel. Because the Ames tunnel is larger, which will further decrease wall effects, the team expects to test using larger, more flight-representative thrust coefficients.

CFD will continue to build validity in code-to-code and code-to-test comparisons as modeling difficulty increases. Some planned milestones include engine startup, six-degree-of-freedom simulations, chemistry effects for reacting nozzle flows, and simulating the Mars atmosphere composed mostly of carbon dioxide.